Ing. Paola Ranut
Prof. Enrico Nobile
Università degli Studi di Trieste
Dipartimento di Ingegneria e Architettura

Corso di Termofluidodinamica Computazionale

Multiphase flows

Examples solved with ANSYS CFX

A.A. 2013-2014

Contents

1 Example: particle tracking

In this Section, an example of particle tracking, solved by ANSYS CFX, is provided. The numerical study investigates a turbulent gas-particle flow in a two-dimensional, vertically oriented backward-facing step. The experimental work of Fessler and Eaton [5] was taken as the reference study for validation purposes. In [5], the measurements of three different gas-particles flows, having different solid particles and mass loadings, are presented.
Other authors performed numerical simulations by using the experimental data of Fessler and Eaton. A Lagrangian approach for predicting the properties of the particle phase was used in the RANS computations of Chan et al. [4] and in the LES simulations of Yu et al. [6]. Benavides and van Wachem [1] analyzed the gas-particle flow field by means of an Eulerian-Eulerian approach.
The computational domain used in the CFD simulations is sketched in Fig. 1. The channel length is $34H$, being $H = 26.7$ mm the hight of the step; the half-width h of the channel upstream the step is 20 mm. In the experimental setup, the inlet channel was 5.2 m long, thus to ensure a fully developed flow at the step. In [1], the length of the inlet channel is assumed equal to $65h$.

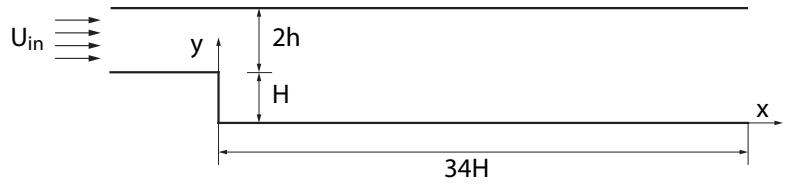


Figure 1: Sketch of the computational domain.

In [2], the hydrodynamic entrance length for a flat duct at Re $= 2.21 \times 10^5$ is estimated as $13.75D_h$, in which D_h is two times the spacing between the duct walls.
In the CFD simulations, in order to have a fully developed velocity profile at the step, two strategies can be followed. In the first one, an extended domain, comprehensive of a sufficient long channel upstream the step, is considered. On the contrary, the second (smarter) choice considers only a short inlet channel, and a full-developed velocity profile is provided at the inlet cross section. This profile can be recovered from another simulation or can be defined by means of analytical expressions. As an example, for the case of fully developed turbulent flow in a flat duct, in [2] the following relations are provided:

$$\frac{u(y)}{u_{max}} = 1 - \frac{n-s}{n-1}\left(\frac{y}{h}\right)^2 - \frac{s-1}{n-1}\left(\frac{y}{h}\right)^{2n} \tag{1}$$

$$\frac{u_m}{u_{max}} = 1 - \frac{n-s}{3(n-1)}\left(\frac{y}{h}\right)^2 - \frac{s-1}{(n-1)(2n+1)}\left(\frac{y}{h}\right)^{2n} \tag{2}$$

in which u_m and u_{max} are the average and the maximum flow velocity, and y is the transverse distance measured from the duct axis. For $4000 \leq$ Re $\leq 10^5$, the following expressions for s and n are provided:

$$s = 0.004 \cdot \text{Re}^{3/4} \tag{3}$$

$$n = 0.00625 \cdot \text{Re}^{3/4} - 2.0625 \tag{4}$$

In the above relations, the the Reynolds number is defined on the mean velocity u_m and $D_h = 4h$, being h the half spacing between the duct walls.

1.1 Definition of the Reynolds number for the backward-facing step problem

The literature offers several definitions of the characteristic length for a backward-facing step problem, which lead to different definitions of the Reynolds number, as stressed in [3]. With reference with the variables reported in Fig. 1, the Reynolds number can be defined as:

$$\mathrm{Re}_{Dh} = \frac{\rho U_b 4h}{\mu} \tag{5a}$$

$$\mathrm{Re}_{2h} = \frac{\rho U_b 2h}{\mu} \tag{5b}$$

$$\mathrm{Re}_{H} = \frac{\rho U_b H}{\mu} \tag{5c}$$

in which U_b is the average velocity of the inlet flow.
Fessler and Eaton [5] provide two other definitions, one for flow in the inlet channel:

$$\mathrm{Re}_{h} = \frac{\rho U_0 h}{\mu} \tag{6}$$

and the other for the flow downwards the step:

$$\mathrm{Re}_{H} = \frac{\rho U_0 H}{\mu} \tag{7}$$

in which U_0 is the centerline velocity of the inlet flow.

1.2 Available data

The experimental data reported in [5] are summarized in Tab. 1 and Tab. 2.

Nominal diameter (μm)	90	150	70
Material	glass	glass	copper
Density ($\mathrm{kg \cdot m^{-3}}$)	2500	2500	8800
Stokes mean particle time constant, $\tau_{p_{Stokes}}$ (ms)	61	167	130
Modified mean particle time constant, τ_p (ms)	38	92	88
Large-eddy Stokes number, St	3.0	7.2	6.9
Particle Reynolds number, Re_p	7.3	11.8	5.5
Mass loading	20%	20%, 40%	3%, 10%

Table 1: Particle parameters.

Centreline velocity, U_0 ($\mathrm{ms^{-1}}$)	10.5
Channel flow, Re_h	13800
Backward-facing step flow, Re_H	18400
90 μm glass velocity ($\mathrm{ms^{-1}}$)	0.46
150 μm glass velocity ($\mathrm{ms^{-1}}$)	0.92
70 μm copper velocity ($\mathrm{ms^{-1}}$)	0.88

Table 2: Fluid parameters.

As the reader certainly knows, the Stokes number St quantifies the nature of the kinetic equilibrium between the particles and the surrounding fluid. For $\mathrm{St} \ll 1$ the particulate phase does

not affect the motion of the fluid phase, and a one-way coupling approach can be applied. Vice verse, for St $\gg 1$ the inertia effect of particles becomes significant, and a two-way coupling technique must be used.
The Stokes number is defined as the ratio of the particle response time τ_p to a representative time scale in the flow τ_f:

$$\mathrm{St} = \frac{\tau_p}{\tau_f} \tag{8}$$

In the case of a creeping flow[1] of solid spherical particles in a gaseous medium, the effect of the fluid density can be neglected and the particle response time can be expressed as:

$$\tau_{p_{Stokes}} = \frac{\rho_p d_p^2}{18\mu} \tag{9}$$

where ρ_p is the particle density, d_p is the particle diameter and μ is the dynamic viscosity of the fluid. However, in the case of no-creeping flows, an alternative definition of τ_p must be used; in [5], the following modified time constant is employed:

$$\tau_p = \frac{\tau_{p_{Stokes}}}{1 + \mathrm{Re}_p^{0.687}} \tag{10}$$

being Re_p the Reynolds number characterizing the particle motion, defined as:

$$\mathrm{Re}_p = \frac{d_p U_{\mathrm{rel}}}{\nu} \tag{11}$$

In the above equation, U_{rel} is a velocity scale which characterizes the average slip velocity of particle relative to the flow.
In [5], the fluid time scale was defined as:

$$\tau_f = \frac{5H}{U_0} \tag{12}$$

1.3 CFD simulation

Benavides and van Wachem [1] employed a $65h$ channel upwards the step, corresponding to a length of 2.6 m. In order to obtain a center-line velocity U_0 of about 10.5 m/s at the step ($x/H = 0$), they imposed a uniform velocity $U_{in} = 9.3$ m/s at the inlet of the channel, corresponding to a Reynolds number of 1.3×10^4. Chan et al. [4] specified, at the inlet of the backward facing step flow, the profiles of velocity previously computed on a 5.2 m long channel flow.
For the particle phase, at the inlet Chan et al. [4] specified, in the streamwise direction, the velocities recovered during the experiment and reported in Tab. 2 (0.88 m/s for the 70µm copper particles), while the transverse velocity components were set to zero. However, Benavides and van Wachem [1] claim that inlet velocity of the particulate phase does not influence the results.
In the present simulation, two different solution strategies are followed. In the first one, an extended geometry is considered, comprehensive of a 3 m inlet channel. The length of the channel was chosen in order to have a fully developed velocity profile at the outlet, and is intermediate between the lengths of [1] and [4]. However, as the post processing confirmed, a slightly shorter inlet channel could have been used.

[1] A creeping flow, also named Stokes flow, is a type of fluid flow in which the inertial forces are small compared with the viscous forces. These flows are characterized by Re $\ll 1$.

In the second approach, the inlet channel is omitted, and only the fluid domain from $x/H = 0$ is considered. The profiles of velocity for the gas and particle phase are initialized with the data computed on a flat duct of height 2h and length 3 m. Here after, this flat duct will be indicated as *initialization duct*.
In order to get Re_h and Re_H as close as to the values reported in [5], both in the extended domain and in the initialization duct, a uniform streamwise velocity of 9.77 m/s was specified at the inlet. The resulting Reynolds numbers are $\mathrm{Re}_h = 13796$ and $\mathrm{Re}_H = 18417$, while the centerline velocity at $x/H = 0$ is $U_0 = 10.66$ m/s. On the contrary, in order to have a centerline velocity $U_0 = 10.5$ m/s, as that reported in [5], a free stream velocity of 9.62 m/s should be applied, which provides $\mathrm{Re}_h = 13587$ and $\mathrm{Re}_H = 18139$. However, the differences between the two cases are negligible and, therefore, in the next sections only the results computed with an inlet velocity of 9.77 m/s will be illustrated.

1.4 Set up of the CFD simulation

1.4.1 Expressions

For simplicity reasons, it is useful to use the *Expressions workspace* to generate and edit expressions via CEL (CFX Expression Language), which can then be used in CFX-Pre in place of almost any numeric value[2].
By double-clicking *Expressions* in the Outline workspace, or by inserting or editing an existing expression, the Expressions workspace opens in a new tab. Then insert the quantities reported in Fig. 2. Be careful to specify the right units of measurement between square brackets, when necessary.

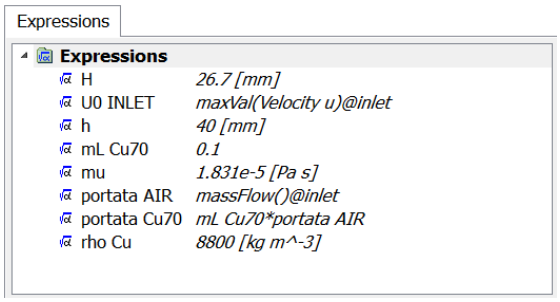


Figure 2: Expressions.

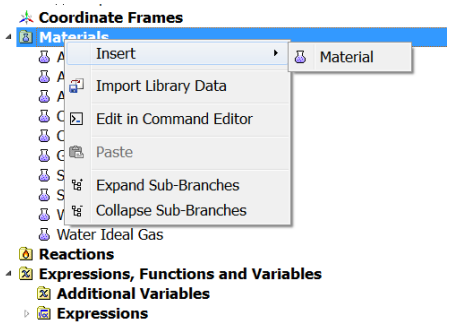
1.4.2 Definition of a new material

Insert the new material Copper70 as illustrated in Fig. 3.

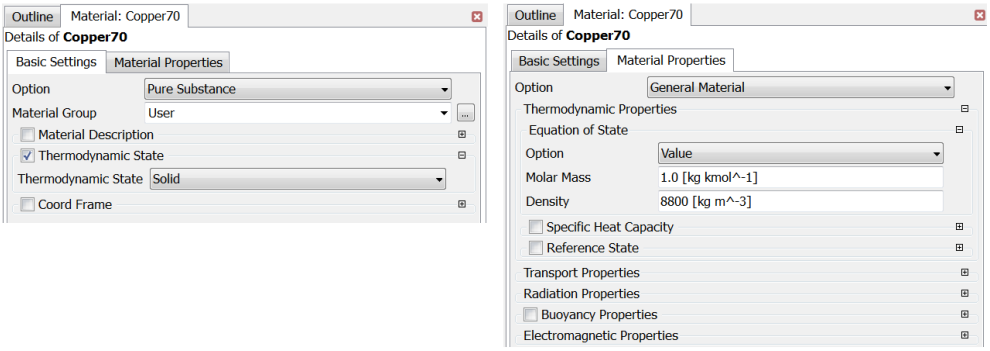
1.4.3 Default domain settings

One fluid and one particle flow must be defined in the *Basic setting* Tab, that is *Air at 25°C* and *Particle*, as illustrated in Fig. 4. For the solid particles, the option *Particle transport solid* must be specified. Gravitational effects can be neglected.
In the *Fluid Models* Tab, select the SST turbulence model and specify the absence of heat transfer. In the *Fluid specific model* Tab, the *Erosion model* and the *Particle Rough model* can be

[2]For further information on the Expressions workspace please refer to Section 28 of the CFX-Pre User's Guide.



(a)



(b)

Figure 3: Definition of a new material.

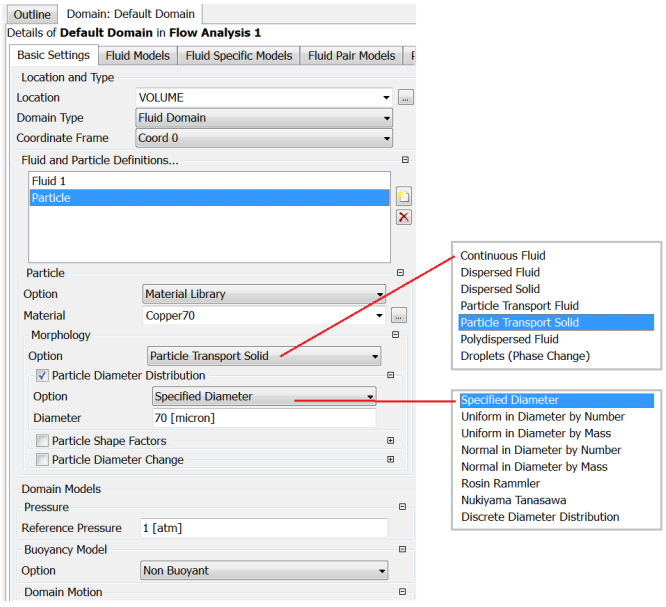


Figure 4: Default domain settings.

defined, but they will be omitted in the present simulation. The *Fluid pair model* Tab is used to specify the kind of interactions between the fluids; the settings used in this simulation are summarized in Fig. 5.

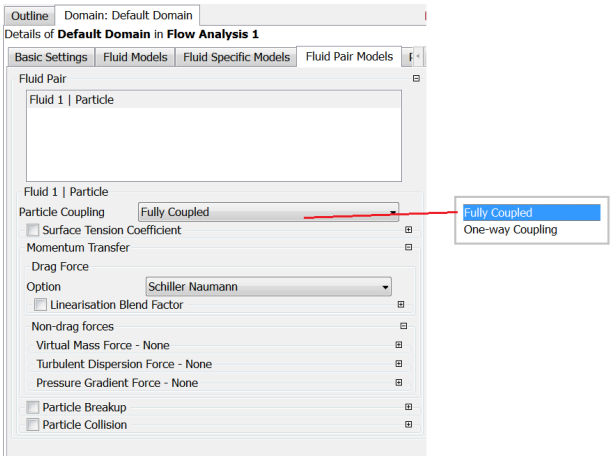


Figure 5: Default domain settings.

In the *Particle injection regions* Tab, injection regions can be employed to define locators anywhere within a domain, and can be set up as spheres, cones, or by using a custom Fortran subroutine[3]. In this example no injection regions are defined.

1.4.4 Boundary conditions: inlet

At the *Inlet*, please set a Normal Speed of 9.77 m/s (or, as an alternative, define U velocity component as 9.77 m/s while keeping $V = W = 0$ m/s). The turbulence intensity can be kept at its default value 5% (it can be seen that the results do not change if the intensity is lowered to 1%).
In the *Fluid Values* Tab, the boundary conditions for each fluid in an Eulerian multiphase simulation and each particle material when particle tracking is modeled can be set[4]. Please refer to Fig. 6 in order to specify the right conditions.

1.4.5 Other Boundary conditions

For the other surfaces the following boundary conditions must be set:

- Outlet → Outlet b.c. → Average static pressure 0 Pa.
- Back, Front → Simmetry b.c.
- Wall → No slip wall b.c.

1.4.6 Output Control

In the *Results Tab*, please select the *Extra Output Var. List* check box, and then select the following additional variables:

- Particle.Average Velocity u
- Particle.Average Velocity v
- Particle.Average Velocity w

[3] CFX-Pre User's Guide, Section 12.4.8. Solver Modeling Guide, Section 8.8.
[4] CFX-Pre User's Guide, Section 14.2.3.

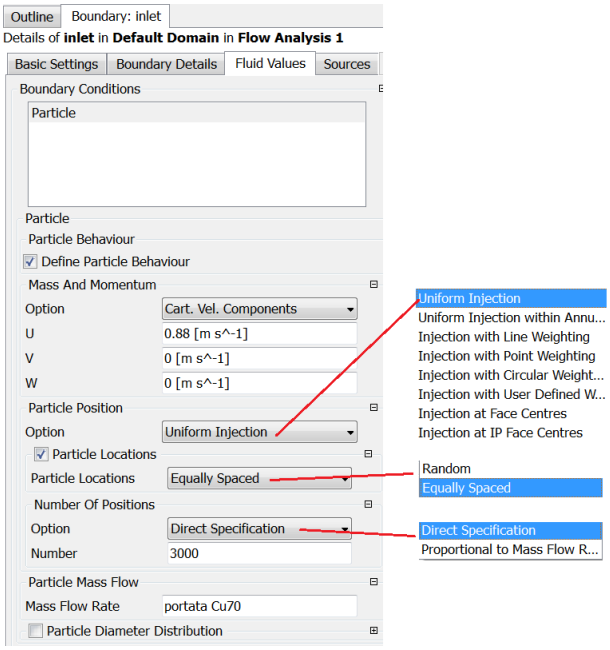


Figure 6: Boundary conditions at inlet.

- Particle.Averaged Particle Number Rate

In order to check whether a steady state solution is obtained, a monitor point can be defined, thus to track the evolution of the velocity in one point of the domain. To do this, go to the *Monitor Tab*, select the option *Monitor objects*, and insert a new monitor point, which can be called *velocity*. Specify the following options:

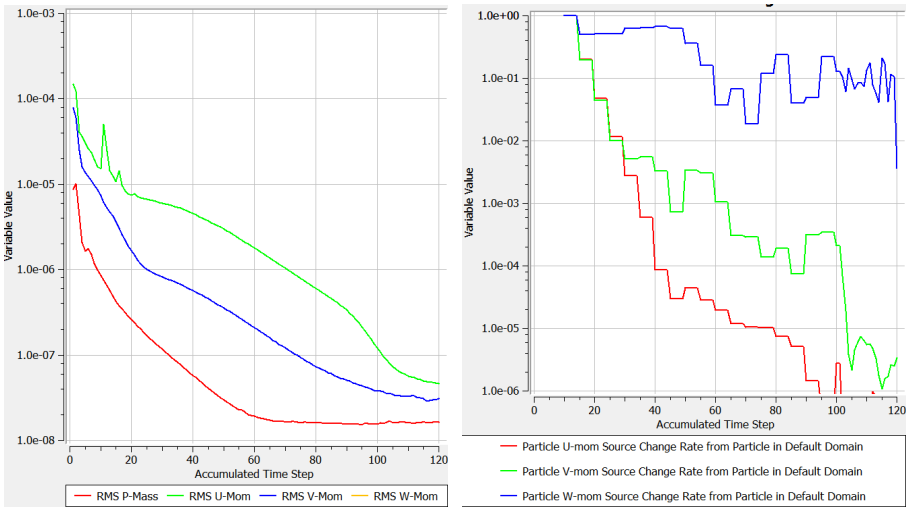
- Option: cartesian coordinates
- Output variable list: velocity
- Cartesian coordinates: 0.3, 0.04, 0

1.4.7 Solver Control

In the *Basic Settings Tab*, please select the *High Resolution* for the advection scheme, while *First order* for the turbulence numerics. The *Fluid timescale control* can be kept at its default values (i.e. *Auto timescale*). Please lower the convergence target to 1e-07, thus to ensure that a steady state solution can be reached.
In the *particle Control Tab*, select the *Particle Termination Control* and specify the following values: *Maximum tracking time* 100 s, *Maximum tracking distance* 100 m, *Maximum number of integration steps* 10000.

1.5 Results

In Fig. 7 the decrease of the residuals, the trend of the particle sources terms and the evolution of the velocity in the monitor point are reported.
In a two-way coupling simulation, the particles are introduced only after a few iterations, thus to allow the continuous phase to settle down from the initial guess. This can be evinced from the evolution of the the particle source term, Fig. 7(b). During the solution, particles are injected at regular iteration intervals: the tracking is made by using the fluid solution field from the previous iteration. Once the particle paths have been calculated, the particle sources to mass and momentum equations are calculated, and subsequently applied to the fluid equations at each subsequent iteration until they are recalculated at the next injection[5].



(a) (b)

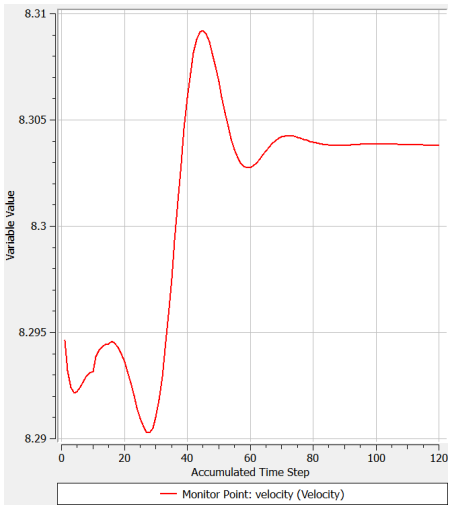


Figure 7: (a) RMS residuals, (b) Particle sources terms and (c) monitored velocity in the point spefied in Section 2.2.5.

In Fig. 8, there is a comparison of the normalized stream-wise gas mean velocity computed with ANSYS CFX with the experimental one of Fessler and Eaton [5] and the RANS results of Chan

[5]CFX Modeling Guide, Section 8.10.1. *Particle Coupling Control.*

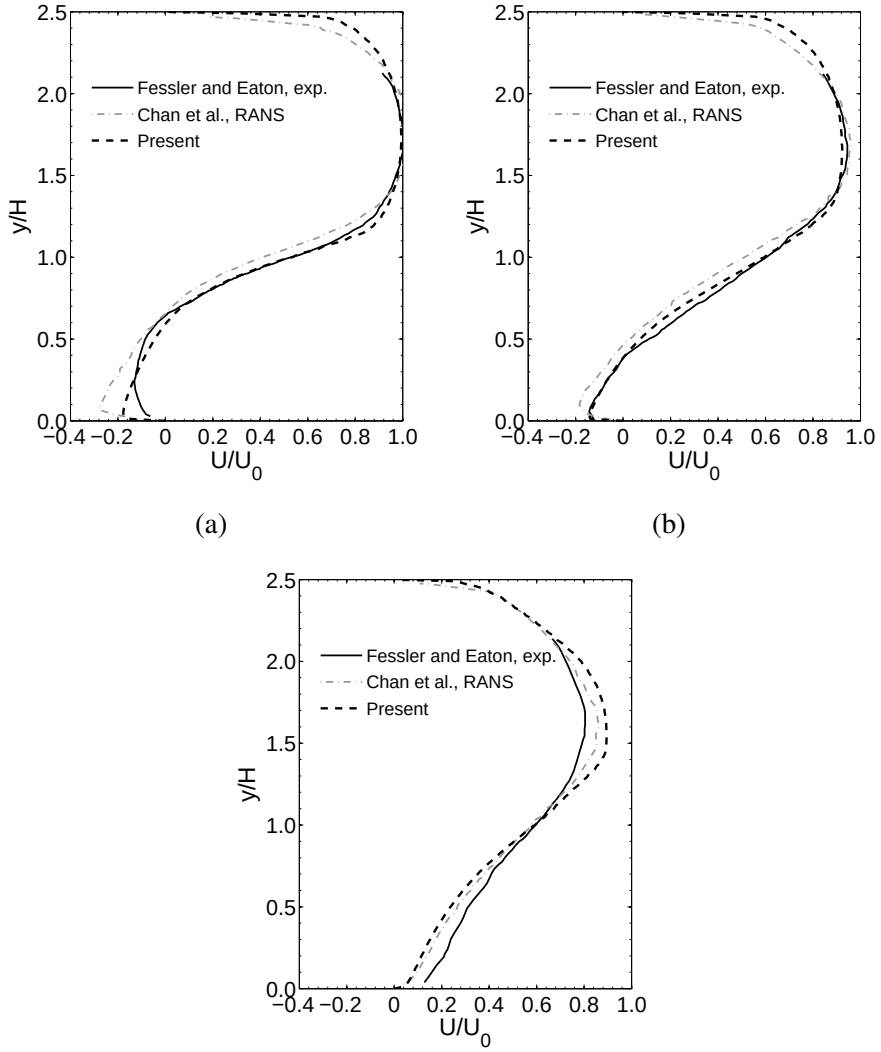


Figure 8: Comparison of the normalized stream-wise gas mean velocity with the values of ref. [5] and [4], for three different cross section (a) x/H=2, (b) x/H=5 and (c) x/H=9.

et al. [4], at three different cross sections, i.e., x/H=2, x/H=5 and x/H=9. As it can be seen, the goodness of the present predictions depends on the cross section considered: in fact, at x/H=9, the present computation overestimates the maxim value of U/U_0.
In Fig. 9 the experimental results of Fessler and Eaton [5], for 70 µm copper particles and two mass loadings, are reported for four different cross sections. As it can be seen, the gas-phase mean velocity profiles are not remarkably changed by particle loading, except at the farthest downstream location, i.e., x/H=14, here omitted. At x/H=2, the particle velocities are lower than the fluid velocities, while downstream the particle velocities exceed the gas velocities due to the deceleration of the fluid in the sudden expansion.
In Fig. 10, the present results are illustrated: the acceleration of the particle phase, with respect to the mean fluid motion, can be clearly evinced.
Figure 11 shows a comparison of the present results for the particulate phase with some published in the literature. As it can be seen, while in proximity of the step the computed profiles

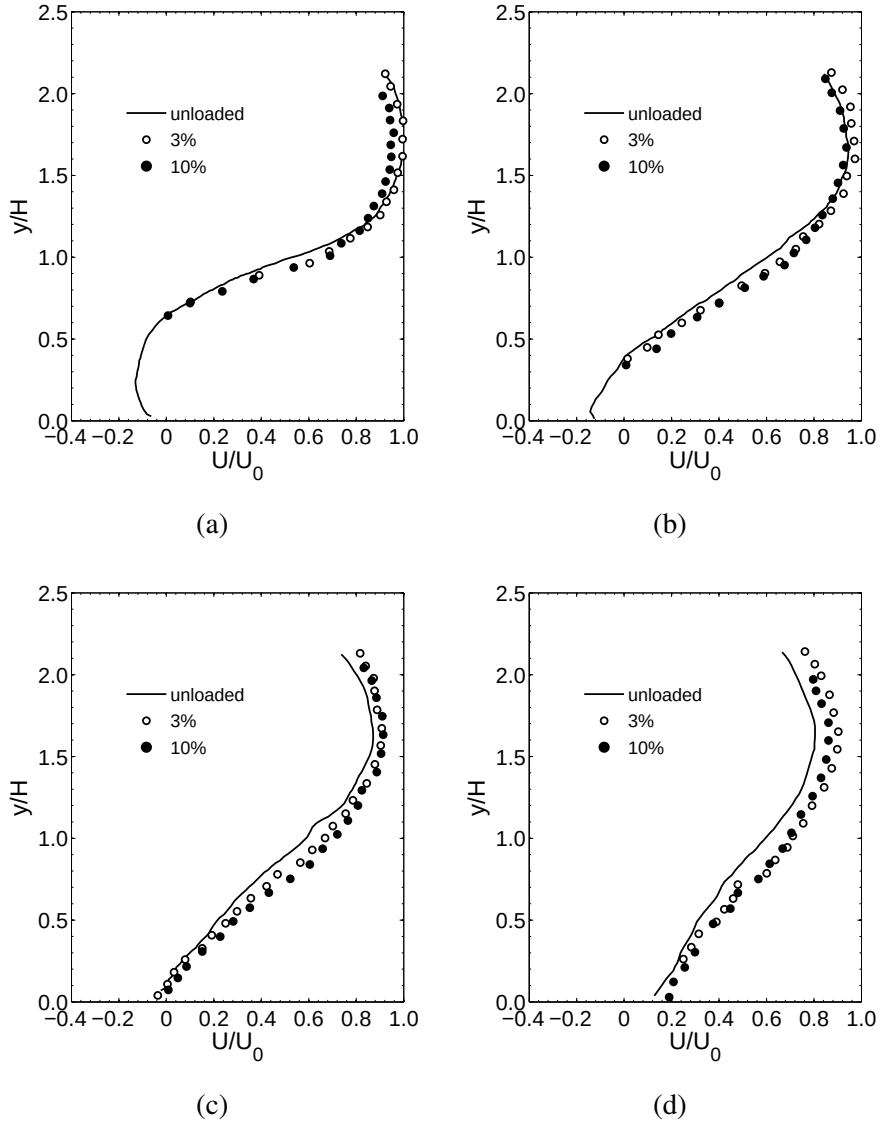


Figure 9: Experimental results of Fessler and Eaton [5], for 70 μm copper particles and two mass loadings, 3% and 10%. (a) x/H=2, (b) x/H=5, (c) x/H=7 and (d) x/H=9.

agree well with the experimental data, downstream the channel the deviation becomes more consistent. Besides the SST turbulence model, to which the previous results refer, two other turbulence models were tested, i.e., the $k-\varepsilon$ and the BSL. In Fig. 13 the effect of the turbulence model on the profiles of the gas phase velocity, at section x/H=5, is illustrated. While in the upper part of the section there is not a great difference among the the velocity profiles computed with the different turbulence models, the SST model seems to predict better the profile in the lower part of the cross section. Furthermore, even the degree of turbulence was tested, but no effect was registered on the results.

In Table 3, the reattachment lengths predicted with the three tested turbulence models are compared with the experimental data [5] and the RANS results of [4]. As it can be seen, the prediction of the SST turbulence model are the closest to the experimental results. This fact encouraged us to present only the results obtained with this turbulence model.

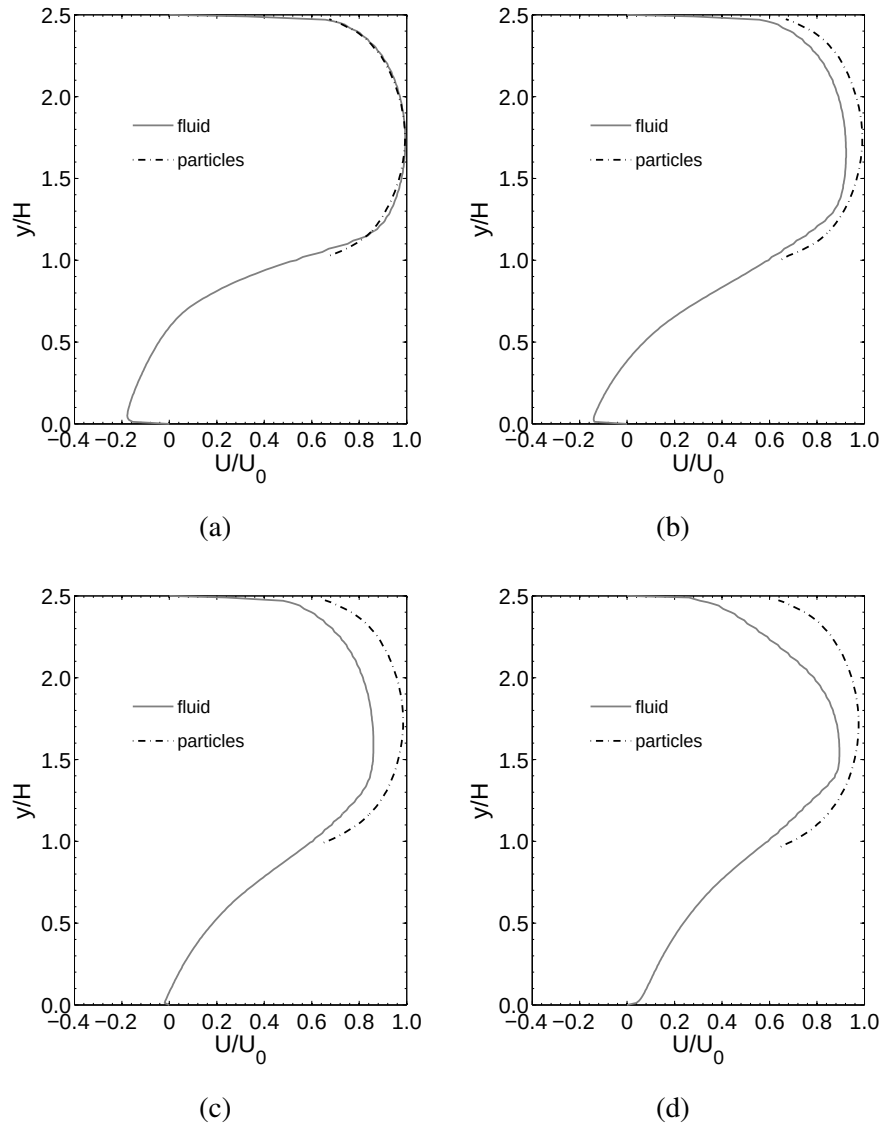


Figure 10: CFX results for 70 µm copper particles at (a) x/H=2, (b) x/H=5, (c) x/H=7 and (d) x/H=9.

A vector plot in proximity of the recirculation region is illustrated in Fig. 14.

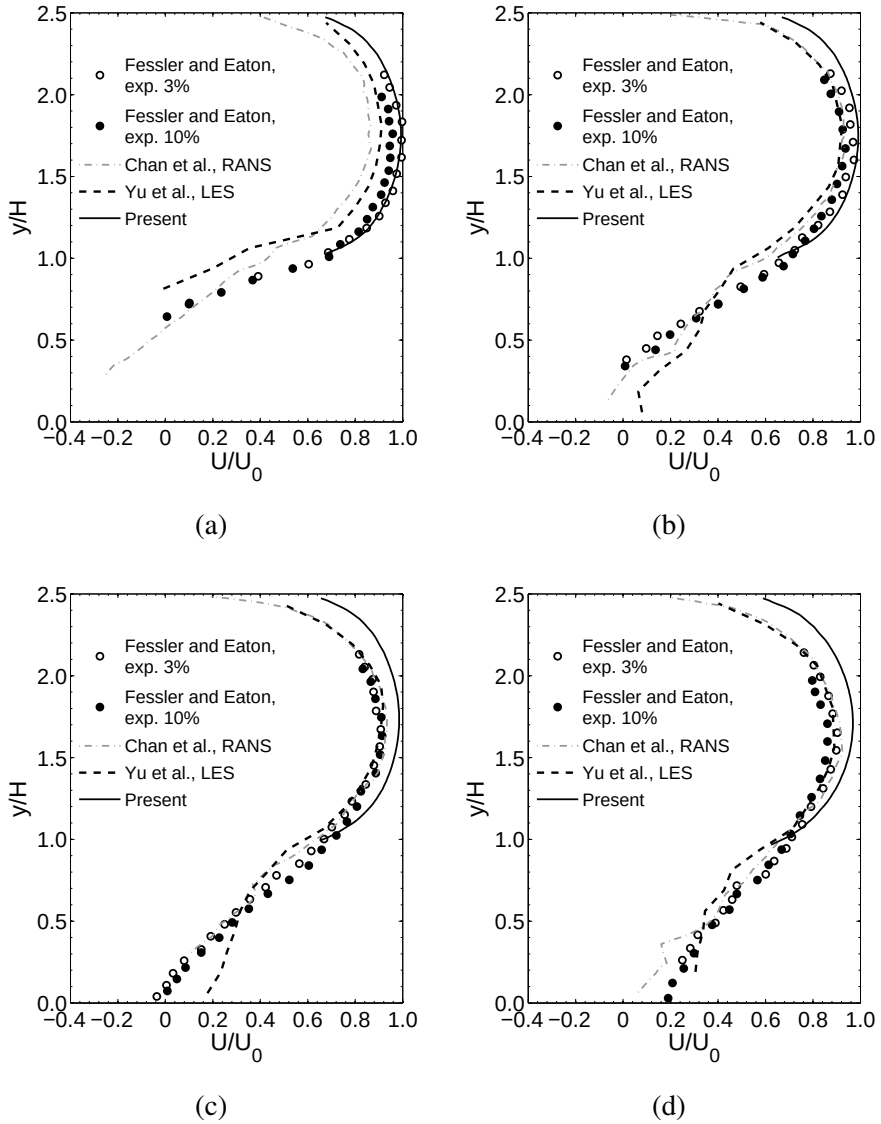


Figure 11: Stream-wise particulate phase mean velocity, for 70 µm copper particles. (a) x/H=2, (b) x/H=5, (c) x/H=7 and (d) x/H=9. Comparison with the results published in the literature.

	x/H
Fessler and Eaton, [5]	7.6
Chan et al. [4]	7.4
Present, SST	7.42
Present, $k-\varepsilon$	6.52
Present, BSL	6.61

Table 3: Prediction of the reattachment length.

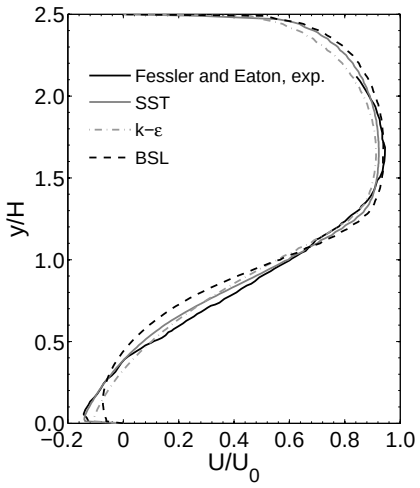


Figure 12: Effect of the turbulence model on the profiles of the gas phase velocity, at section x/H=5.

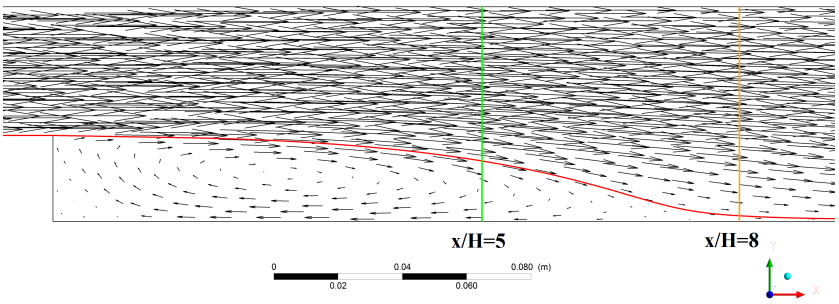


Figure 13: Vector plot in proximity of the recirculation region

1.6 Comments

1.6.1 Effect of the number of particles

With a higher number of particles the result is expected to be smoother but, at the same time, the computational burden increases. Therefore, the proper number of particles to be used should be checked through a sensitivity study, similar to the mesh dependency tests.
For this tutorial, 500 and 3000 particles were tested: the results demonstrated that 500 particles are not enough to get reliable statistics. In fact, as it can be seen from Fig. 15, the profile of the particle phase velocity for 500 particles is not smooth but presents some abrupt variations. However, it is worth pointing out that, despite the number of particles used, no particles are

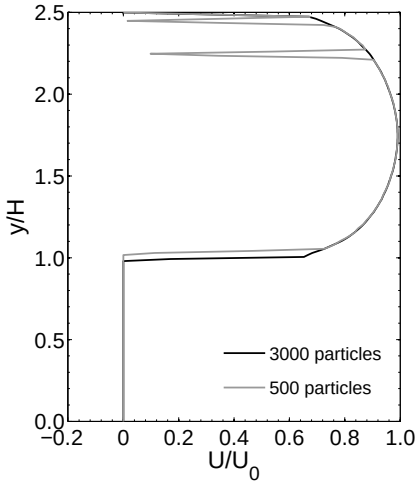


Figure 14: Effect of the number of particles on the profiles of the normalized particle phase velocity (Particle.Averaged Velocity u/U0), at section x/H=5.

found in the lower part of the domain, unlike what happens in the experimental tests and in the numerical results of Chan et al. [4]. This fact may be connected to a limit of the CFX Solver, since no effect was registered by varying the king of particles' injection.

1.6.2 Effect of buoyancy

The effect of buoyancy was tested by specifying, along the x direction, a gravitational acceleration equal to g, while keeping the other components null. This setting was done according to the orientation of the domain in the experimental set up.
As Fig. 16 confirms, the results of the buoyant simulation worsen with respect to the previous ones, since the velocity of te particle phase increases.

1.6.3 Velocity profile from expression

As said before, as an alternative to the use of the initialization duct, a velocity profile can be specified in correspondence of the step, i.e., the section at $x/H = 0$. Generally, this kind of profiles can be recovered from experimental data (if available) or from existing CFD simulations, previously run on proper domains. In our case, the initialization data were exported from a simulation on a duct channel of 3 meters length.

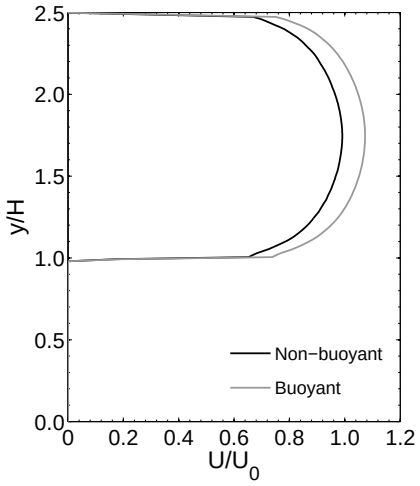


Figure 15: Effect of the buoyancy in x direction on the profiles of the normalized particle phase velocity (Particle.Averaged Velocity u/U0), at section x/H=5.

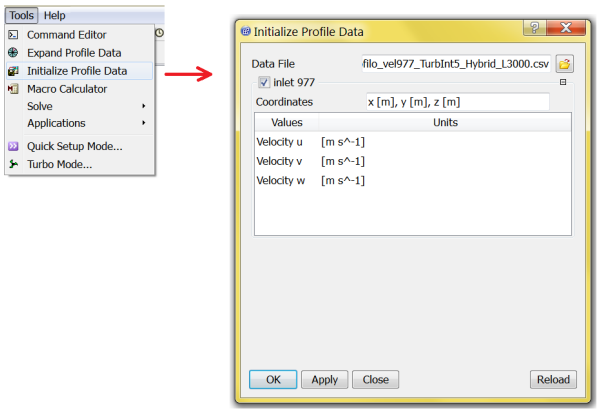


Figure 16: Initialize profile data.

	A	B	C	D	E	F	G	H	I	J	K
1	[Name]										
2	inlet 977										
3											
4	[Spatial Fields]										
5	x,y,z										
6											
7	[Data]										
8	x [m], y [m], z [m], Velocity u [m s^-1], Velocity v [m s^-1], Velocity w [m s^-1]										
9	0.00000000e+000, 6.66999966e-002, 0.00000000e+000, 0.00000000e+000, 0.00000000e+000, 0.00000000e+000										
10	0.00000000e+000, 6.63999990e-002, 0.00000000e+000, 5.76503801e+000, 1.94707457e-002, -2.16840434e-019										

Figure 17: Structure of the initialization file .csv.

The specification of the velocity profile please refer to Fig. 17 and following. Go to *Tools* and select *Initialize Profile Data*, then a dialog box appears. Hence select the file containing your profile data, and click *Open*. The profile data is loaded and the profile data name, coordinates, variable names and units are displayed. The .csv file containing the initialization values must be organized as illustrated in Fig. 18: a name must be provided for this profile (*inlet 977* in this case).
Under the library section of the object tree, a new *User Function* object is generated for this

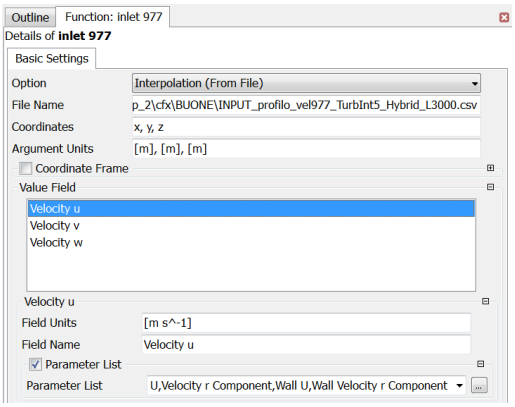
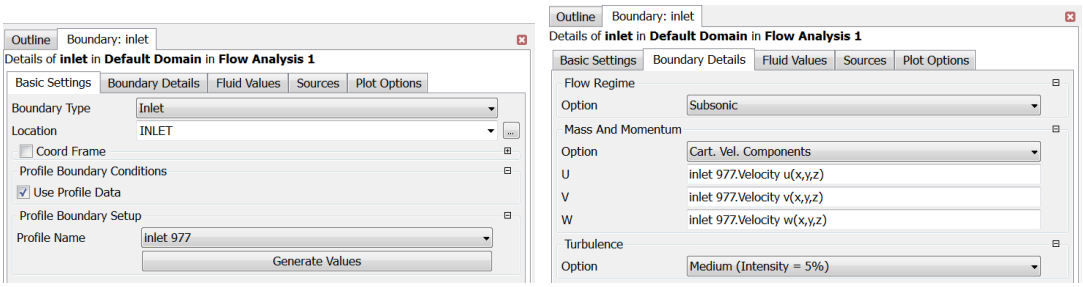


Figure 18: The generated *User function.*



(a) (b)

Figure 19: Initialize profile data: *Inlet* boundary condition.

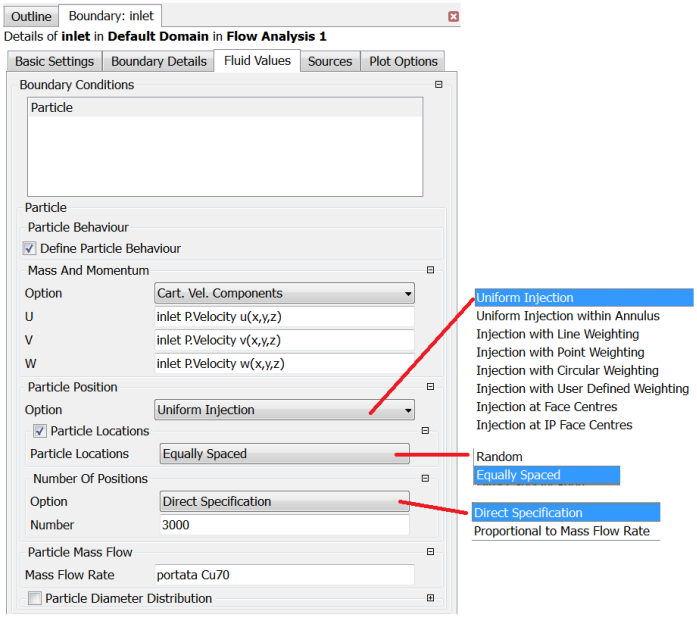


Figure 20: Initialization for the particulate phase.

profile function, as can be seen from Fig. 19, whose name is the one provided in the .csv file. In order to effectively use the desired profile file, the user must go to the *Inlet* boundary condition and activate the option *Use profile data*, see Fig. 20. Then the desired profile must be selected (if more then one profile has been uploaded) and activated by clicking on *Generate values*. Now, in the *Boundary details* Tab, the velocity components for the system will be indicated as *function_name*.Velocity(x,y,z) and so on, where *function_name* is the name of the user function generated.
Besides the initialization of the gas phase, also the particulate phase has been initialized in a same manner. The only difference is that the velocity components at the inlet must be specified manually, as illustrated in Fig. 21. In Fig. 21 it was chosen to inject 3000 particles in an uniform way from equally spaced injection points.
As it can be seen from Fig. 22,the velocity profile of the gas phase obtained with the simulation without the initialization duct do not coincide with the the one computed on the other domain (the maximum velocity is slightly overestimated. This fact might be connected to the interpolation of the velocity profile, even thought the node distribution along the y axis should correspond. For this reason, the simulation with the initialization duct was chosen.

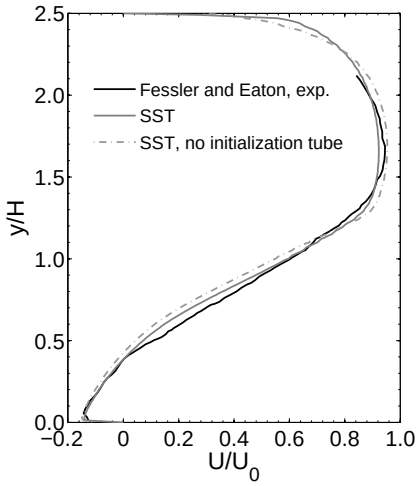


Figure 21: Effect of the type of simulation, with or without the inlet duct, on the velocity profile of the gas phase at x/H=5.

1.6.4 Velocity profile from file

In many cases, a mathematical relationship for the velocity profile is available, e.g., for a developed flow in a circular duct. In this case, this expression can be defined in the *Expressions* workspace, and then recalled when specifying the boundary conditions at the inlet.

1.7 Suggested exercises

Possible suggestions for additional work:

- See how the agreement with the available experimental data can be improved
- Check the effect of the number of particles on the results
- Try to have tracking particles also in the lower part of the domain

- Solve the problem by assuming a one-way coupling instead a two-way one, thus to verify whether the results change and what it is the effect on the computational time
- Solve the problem in an Eulerian-Eulerian framework

2 Example: collapse of a water column (dam breaking problem)

In this Section, a practical example of free surface flow, solved by ANSYS CFX, is provided. The test case considers the collapse of a water column in a tank, having an obstacle placed at the centre of the bottom boundary. A sketch of the computational domain is reported in Fig. 23. The test setup consists of a column of water at rest, located behind a membrane on the left side of a tank. At time $t = 0$ s, the membrane is removed and the water column collapses and flushes to the right due to the large pressure difference between the water and air at the interface. During the impact with the obstacle, a complicated flow structure results, including several captured pockets of air.
For this test case, gravitational acceleration is the driving force causing the water column to seek the lowest possible level of potential energy. At the beginning, the flow is dominated by inertia forces with viscous effects increasing rapidly as the water comes to rest.

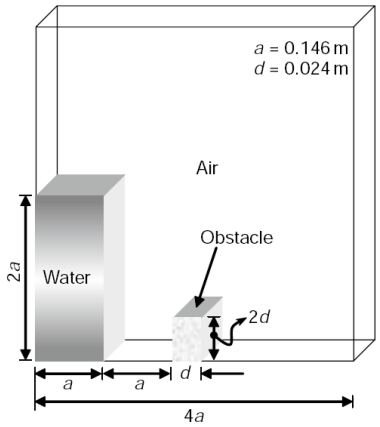


Figure 22: Computational domain employed for the dam breaking problem.

2.1 Computational mesh

The computational structured grid was generated with ANSYS ICEM. Due to the simple geometry, the two-dimensional domain was defined directly in ICEM, through the insertion of points and connecting lines. After the generation of the 2D mesh in the xy plane, the mesh was extruded for 2mm along the z direction, thus to produce a 3D computational domain.
The block partition and the final mesh are reproduced in Fig. 24, while the parameters employed for the definition of the mesh are summarized in Table 4. In Fig. 25 the different parts created are illustrated.
The mesh consists of 17304 nodes and 8444 elements: as it can be seen, in the case of a Cartesian structured mesh, the number of nodes is almost double of the number of elements.

2.2 Set up of the CFD computation

The mesh, saved in format .cfx5, was imported in ANSYS CFX Pre.
Since no mass transfer between the phases occurs, surface tension don't need to be modeled.

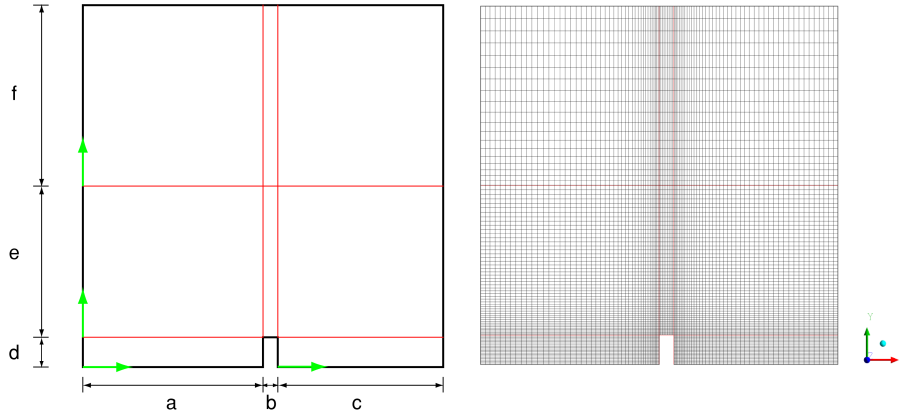


Figure 23: Block-partition of the computational domain (left) and structured mesh employed in the simulations (right).

Edge	Distribution	s_1	s_2	Nodes
a	Poisson	0.0075	0.002	45
b	Uniform			10
c	Poisson	0.002	0.0075	45
d	Uniform			22
e	Poisson	0.002	0.0075	50
f	Poisson	0.0075	0.02	22

Table 4: Parameters for the definition of the mesh.

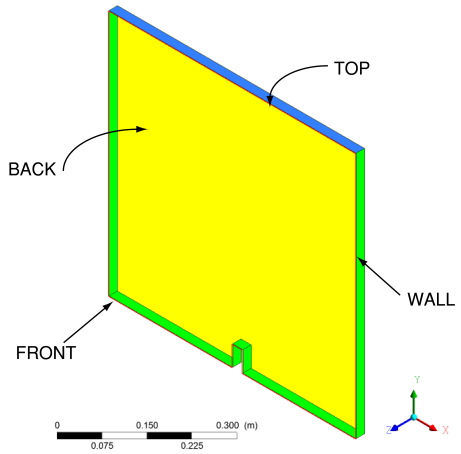


Figure 24: Subdivision of the domain into parts.

2.2.1 Expressions

Simulation of free surface flows usually requires defining initial conditions to set up appropriate volume fraction fields. As it described later, these conditions can be defined through the creation of expressions using CEL (CFX Expression Language). Before doing so, it may be useful to define the initial dimensions of the water column. Therefore, in the *Expressions workspace* two simple expression can be defined. By double-clicking *Expressions* in the Outline workspace, the Expressions workspace opens in a new tab. Then Then,

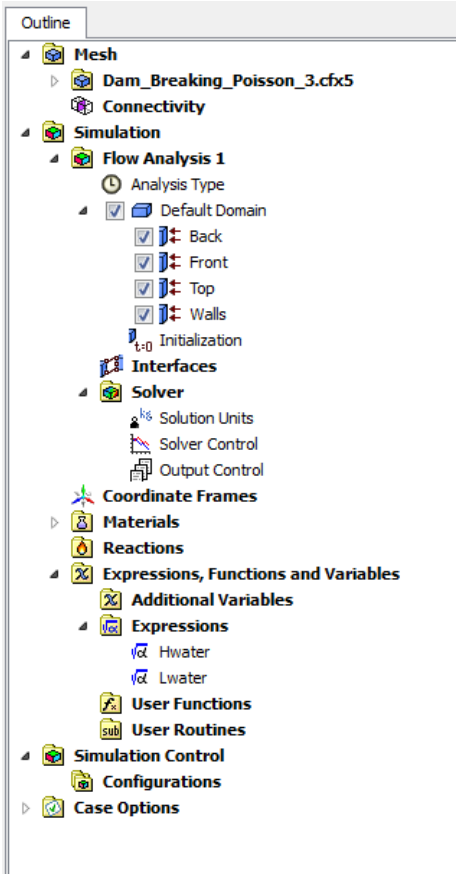


Figure 25: Outline of ANSYS CFX Pre.

- right-click and select *Insert* → *Expression*;
- specify the name for the first variable (i.e., the height of the water column): *Hwater*;
- click *OK*;
- in the lower *Definition Tab*, write the following expression: *0.292 [m]* and click *Apply*;
- repeat the sequence for the width of the water column, calling this variable *Lwater*;
- define *Lwater* as *0.1461 [m]* .

In the remaining part of this Section, the settings of the *Flow Analysis 1* workspace will be illustrated.

2.2.2 Analysis Type

Since the aim of this simulation is the study of temporal evolution of the collapse of a water column, the simulation must be unsteady. Therefore, the time duration and the time steps must be specified, as illustrated in Fig. 27. The choice of using a time step of 0.001 s was enough precautionary, thus requiring a significant computational time: therefore, in order to reduce the computational burden, a time step of 0.0025 s could be sufficient.

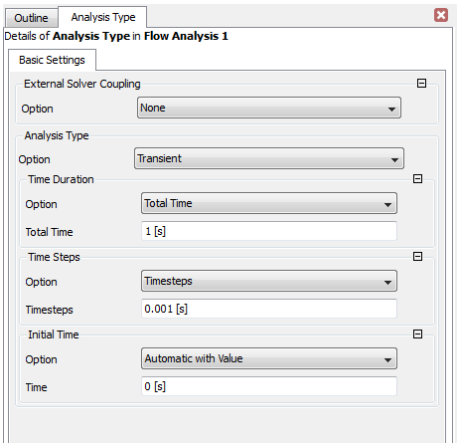


Figure 26: Settings for Analysis type.

2.2.3 Domain: Default Domain

Basic Settings Two fluids must be defined, that is *Air at 25C* and *Water*, as illustrated in Fig. 28. For both fluids, the option *Continuous Fluid* must be specified.
The collapse of the water column is driven by gravitational forces: therefore, a buoyant model must be considered. In reference to the coordinate system defined in Fig. 25, the gravitational force is directed along the y axis.
The reference density for buoyancy is that of the lighter fluid, that is Air.

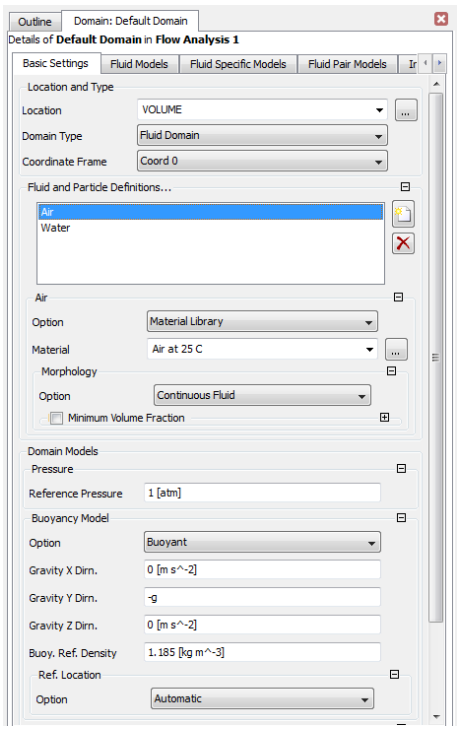


Figure 27: Settings for Domain: Default Domain - Basic Settings.

Fluid Models The Fluid Models tab is where models for the Eulerian flow field are chosen. There are two models available for free surface flow: homogeneous and inhomogeneous. As already said, the homogeneous model is a special case of the mixture formulation, valid when the different phases share the same velocity field. The homogeneous model can be used when the interface between the two phases remains well defined and none of the dispersed phase becomes entrained in the continuous phase.
The settings for this Tab are summarized in Fig. 29 and refer to the Homogeneous Model. Usually the predictions of an inhomogeneous flow model are more accurate, but are more computationally expensive, since it is necessary to solve the flow fields of both phases.

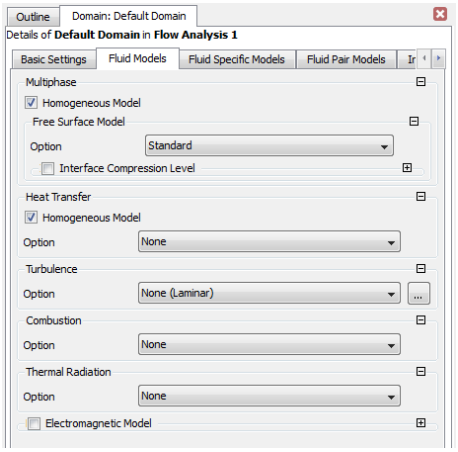


Figure 28: Settings for Domain: Default Domain - Fluid models.

The modeling of the free surface was performed through the *Standard free surface model*: in this way, a distinct interface between the fluids can be modeled.

Fluid Specific Models In this Tab, the buoyancy models for the Air and Water phase must be selected. In this case, since no heat transfer occurs, only the *Density Difference Model* is allowable[6].

Fluid Pair Models This Tab is allowable only in the case of multiphase simulations, and is used to specify the kind of interactions between the fluids. The settings for the Fluid Pair Models Tab are illustrated in Fig. 30. In this case, a surface tension coefficient, equal to 0.07 N/m, is specified[7], while no interphase momentum transfer, nor mass transfer[8] is considered.

Initialization At $t = 0$ s, both phases are at rest: therefore, a null velocity field must be initialized (Initial Condition → Cartesian Velocity Components → Automatic with Value).
Moreover, since at the beginning of the simulation the water column occupies a well defined region, a distribution of the volume fractions inside the domain must be provided, as summarized

[6]The other model implemented in ANSYS CFX is the Boussinesq Model, which is employed for modeling natural convection effects within a fluid.

[7]Probably, the surface tension effects could have been omitted.

[8]Since there is neither phase change nor breakup/coalescence phenomena, an interphase mass transfer does not takes place.

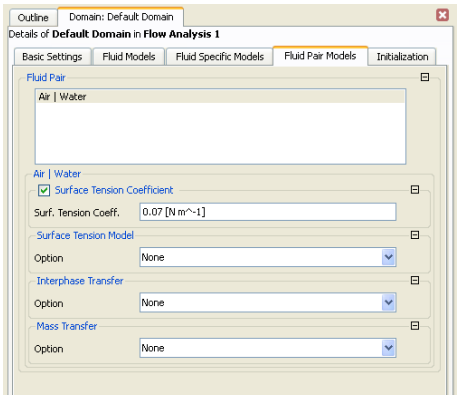


Figure 29: Settings for Domain: Default Domain - Fluid Pair Models.

in Fig. 31. This is done by specifying a logic expression as it follows:

$$if((x < Lwater)\&\&(y < Hwater), 0, 1) \quad \text{for Air} \tag{13}$$

$$if((x < Lwater)\&\&(y < Hwater), 1, 0) \quad \text{for Water} \tag{14}$$

The statement of a logic if condition is the following:

$$if(condition, value\ if\ it\ is\ true, value\ if\ it\ is\ false) \tag{15}$$

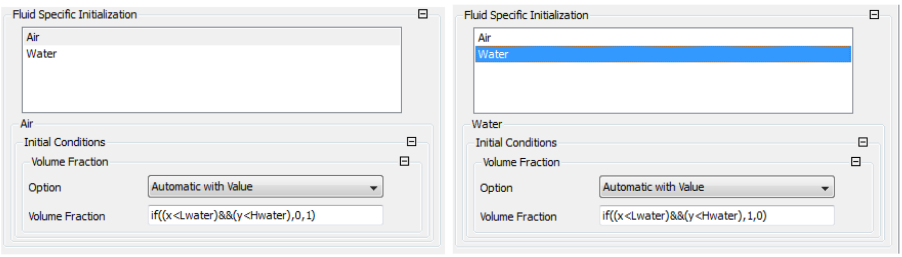


Figure 30: Settings for Domain: Default Domain - Initialization - Fluid Specific Initialization

2.2.4 Boundary conditions

With reference to Fig. 25, the following boundary conditions were applied:

- Front: simmetry;
- Back: simmetry;
- Walls: no slip wall;
- Top: opening[9]
  - Boundary details: Flow regime: subsonic; Mass and Momentum: Entrainment, Relative Pressure: 0 Pa

[9]The *opening boundary condition* allow fluid to exit and enter the computational domain

– Fluid Values: Air, volume fraction: option: value, 1; Water, volume fraction: option: value, 0.

The setting of an *opening boundary condition* allow to free from the interaction of the water with the upper wall (if some particles reach the top boundary, they escape from the domain).

2.2.5 Solver control

For the spatial discretization of the advection term the High Resolution scheme is adopted, while a second-order backward Euler time stepping is applied for the temporal discretization. The Second Order Backward Euler scheme is an implicit time-stepping scheme, which is second-order accurate, and is the default in ANSYS CFX.
At each timestep in a transient simulation, the CFX-Solver performs several coefficient iterations or loops, either to a specified maximum number or to the predefined residual tolerance. The use of a large number of coefficient iterations for transient runs is not recommended: an improved accuracy can be much more efficiently achieved by reducing the timestep size. Anyway, a compromise between timestep size, number of coefficient iterations and residual tolerance must be made thus to get the most cost effective solution.
The settings for the Solver Control are summarized in Fig. 32.

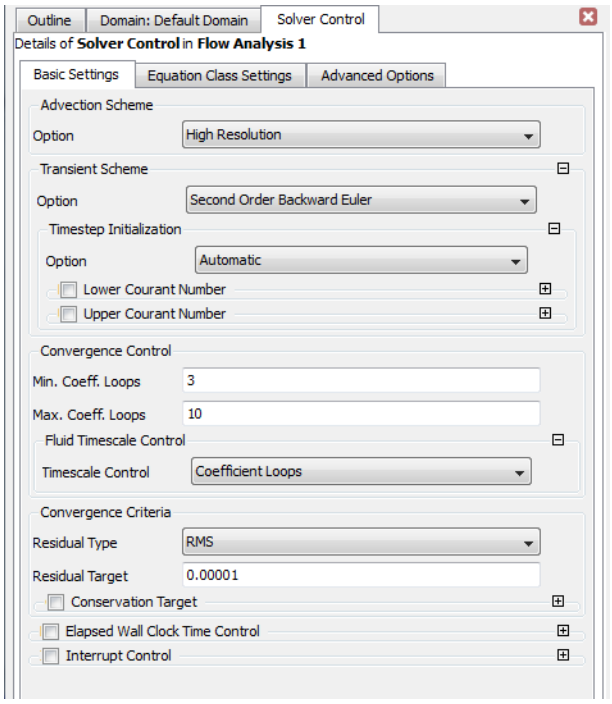


Figure 31: Settings for Solver Control.

2.2.6 Output Control

In the *Transient (Trn) Results Tab* of the Output Control Panel, the output frequency for the save of transient results files can be specified. This is necessary for specific post processing functions (i.e., for creating an animation).
Figure 33 reports the settings for a save of a transient result file every 5 time steps.

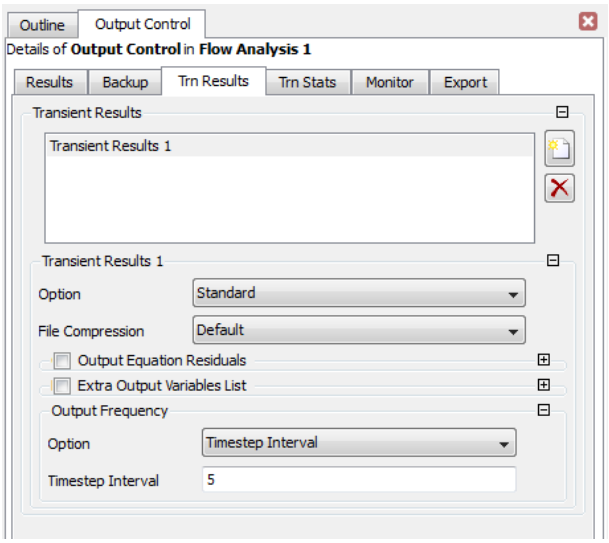


Figure 32: Settings for Output Control - Transient Results.

2.3 Numerical results

To load the result saved for a specified time step, click on the  button and select the desired time frame.
The numerical results predicted by the homogeneous model for a time step of 0.001 s are illustrated in Fig. 34. In this case, the contours of the *Water.Volume Fraction* are illustrated: the number of contours has been set to 15, and the color map to *Inverse Grayscale*. In Fig. 35 and 38 a comparison with two numerical works published in the literature is provided.

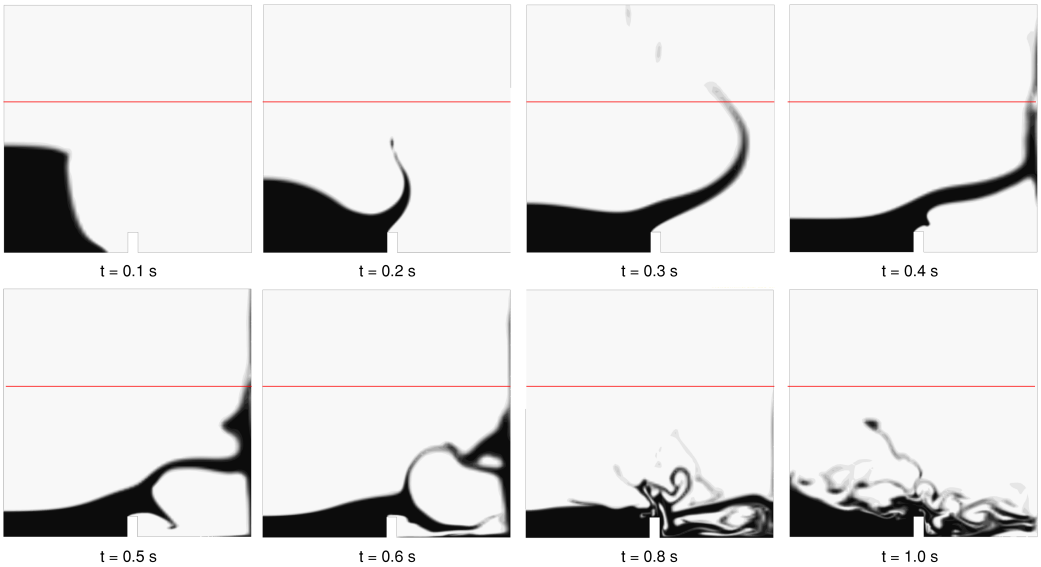


Figure 33: Numerical results of the collapsing water column predicted with the homogeneous model (surface tension coefficient 0.07 N/m, time step = 0.001 s). The red line corresponds to the heigh of the images in Fig. 35 and 38.

As an alternative, only the outline of the Water.Volume Fraction can be plotted, as illustrated

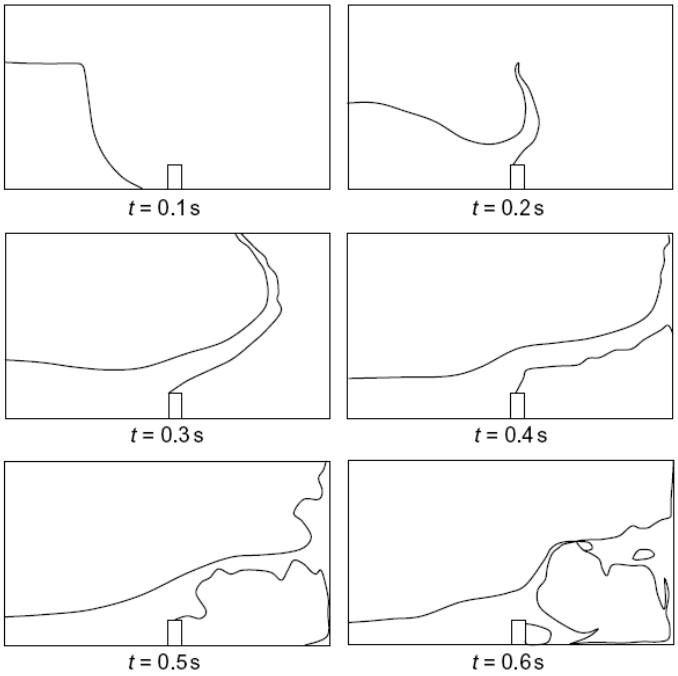


Figure 34: Numerical results of the collapsing water column with an obstacle predicted by Ubbink (1997).

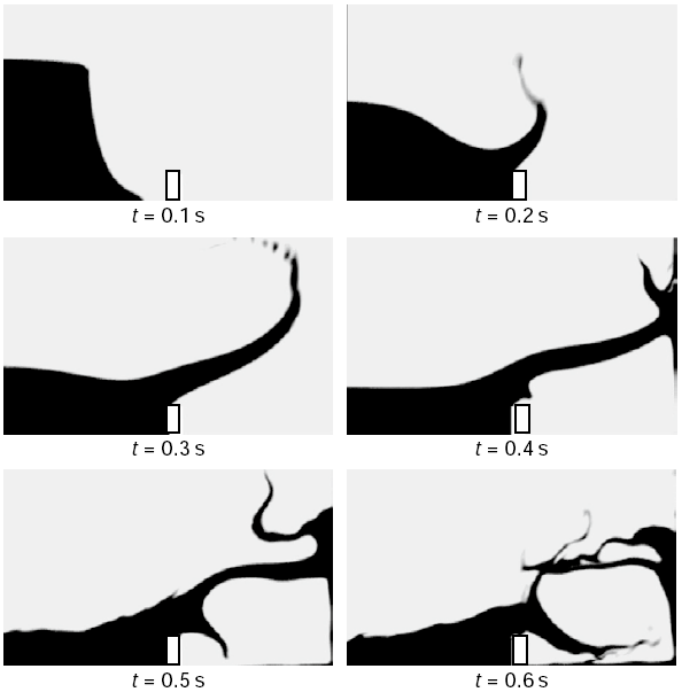


Figure 35: Numerical results of the collapsing water column with an obstacle predicted by Yeoh and Tu (2010).

in Fig. 37. In this case, in the *Geometry* Tab, the option *Value list* must be selected for the *Range*, and the values 0, 0.5, 1 must be specified. Moreover, in the *Render* Tab, the option *Show*

Contours Bands must be unchecked. To draw the contour with line of a specified color, expand the *Show Surface Lines* options, and check the option *Constant Coloring* → *User Specified*.

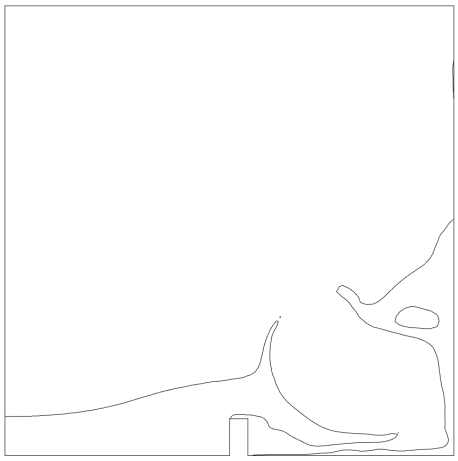


Figure 36: Numerical results of the collapsing water column predicted with the homogeneous model (surface tension coefficient 0.07 N/m, time step = 0.001 s). Result at time 0.6 s.

For this tutorial, it is interesting to create an animation of the evolution of the phenomenon. For doing this, click on the  button. There are two ways for creating a movie, that is *Quick Animation* and *Keyframe Animation*. The former generates an animation from the whole set of the temporary files saved, while the latter allows the user to employ a reduced set of temporary files.
The settings for the *Quick Animation* are illustrated in Fig. 38 and summarized in the following itemize:

- Velocity of reproduction: you can move the cursor between Fast and Slow. Since the phenomenon is enough quick, a slow velocity is preferable (otherwise it could be difficult to capture the characteristics of the phenomenon).
- Loop-Bounce: the *Loop* option allows the user to specify the number of times the movie should be repeated. It is worth noting that the set of transient files is loaded as many times as the number specified in the *Repeat* box.
- For saving the movie, please check the *Save Movie* box and specify the destination folder.
- Select the MPEG1 format for the save.
- To start the recording, push the  button.

Animations can also be based on keyframes. In this case, the start and end frames can be selected, and the number of intermediate frames specified: if the intermediate frames were not saved during the run, then they will be computed by interpolating the existing frames.

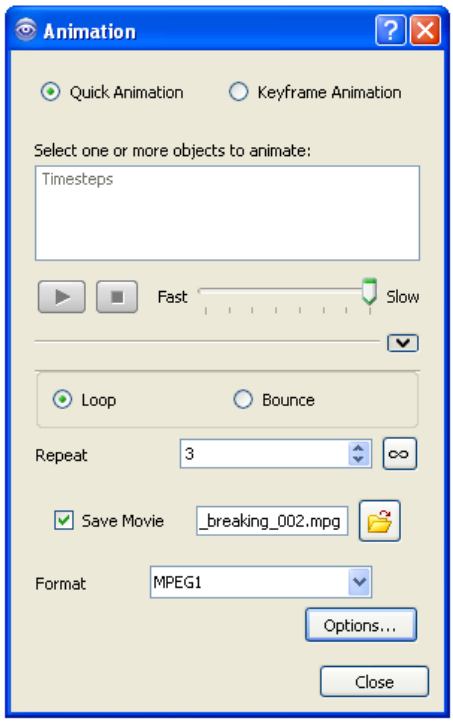


Figure 37: Creation of an animation: settings for *Quick Animation*.

References

[1] A. Benavides and B. van Wachem. Eulerian-eulerian prediction of dilute turbulent gas-particle flow in a backward-facing step. *Int. J. Heat Fluid Flow*, 30:452–461, 2009.

[2] M.S. Bhatti and R.K. Shah. Turbulent and transition flow convective heat transfer in ducts. In S. Kakaç, R.K. Shah, and W. Aung, editors, *Handbook of Single-Phase Convective Heat Transfer*. John Wiley & sons, 1987.

[3] G. Biswas, M. Breuer, and F. Durst. Backward-facing step flows for various expansion ratios at low and moderate reynolds numbers. *J. Fluids Eng.*, 126:362–374, 2004.

[4] C.K. Chan, H.Q. Zhang, and K.S. Lau. Numerical simulation of gas-particle flows behind a backward-facing step using an improved stochastic separated flow model. *Computational Mechanics*, 27:412–417, 2001.

[5] J.R. Fessler and J.K. Eaton. Turbulence modification by particles in a backward-facing step flow. *J. Fluid Mesh.*, 394:97–117, 1999.

[6] K.F. Yu, K.S. Lau, and C.K. Chan. Numerical simulation of gas-particle flow in a single-side backward-facing step flow. *Journal of Computational and Applied Mathematics*, 163:319–331, 2004.